TAMPEREEN AMMATTIKORKEAKOULU

NX- SIEMENS IMPLEMENTION AND PILOTING

ABSTRACT

Tampereen Ammattikorkeakoulu
Tampere University of Applied Sciences
Degree Programme Mechanical Engineering

ALTÉS ANGEL, FRANCESC
NX-CAD Implementation and Piloting

Bachelor's thesis 43 pages
May 2017

The necessity of this project came when TAMK got a license for NX-Siemens software. As all the software, it requires some time and patience at beginning to get familiarize with the first basic tools. In order to make this time more pleasant, it was decided to create some kind of resource to help the interested ones in it.

The purpose, then, was creating a basic explanation in an instruction format. As NX is extremely complex, it was chosen to focus on structural and thermal analysis as are the more general problems related to these kind of problems.

The methodology was organized in the following planning task. First, came the theoretical part where was explained some basic concepts, as finite elements method, that should be known beforehand. After, it came the main part of the thesis which was doing a research in NX-Siemens in order to get familiar with the specific basic tools and dominate both type of analysis to interpret the results that were obtained. Finally, it came the part to organize all the results and knowledge acquired during the research to write a properly basic instructions for NX.

The instruction part is composed by two basic examples, one for each kind of analysis, which are explained step by step the way to solve them with NX always followed with pictures and commentaries that makes it easy to follow.

To conclude, even it is easy to get some basic knowledge through this project about NX, more work will need to be done to explain more of the endless options and tools this software is capable of. Also, it would be interesting to compare NX with other similar software in order to see the clear benefits from it.

Key words: NX-Siemens, finite elements method, structural analysis, thermal analysis

CONTENTS

GLOSSARY

TAMK	Tampere University of Applied Sciences
NX	NX: Siemens PLM software
ANSYS	ANSYS, inc. software
FEM	Finite elements method

1 INTRODUCTION

Nowadays the industrial world is composed by companies which are continuously creating or improving their products to make them more competitive in the market. In software's world is exactly the same. Every year companies create, improve and update new software that do similar or exactly the same tasks.

In the enginery world, one of the first dilemmas before facing a real problem is choosing the best software to solve it in the best efficiency way. Sometimes it is easy making that choice because it will be chosen the software in which one is more familiarized and the one more focused on that kind of problem. Even though, the decision of the dilemma could be more difficult sometimes, maybe because it is faced an unusual problem or also because there is not any familiar software available at that moment. So, having lot of different software to choose it is not as good as it sounds by the reason that every software is a different world. Although it can be two software which do exactly the same, the way them work is completely different most of the times, so it is need some practice and some skills before being able to use them fluently. That is why it will be necessary to be fluently with the more software it is possible in order to solve all the problems with the most effective and faster way.

Focusing more now on the topic of the thesis, it is going to be started with the necessity of it. Few months ago, TAMK University got a new license for the "NX-SIEMENS" which is an advanced high-end CAD/CAM/CAE software. As it is said in the introduction, is so important to use fluency as much software as it is possible, so it is going to be interesting for the school teaching to its students how to use NX. In order to that, the purpose of this thesis will consist of creating an easy instructions to help them getting familiar easily.

As any other project, it will be divided in a theoretical part, which will include a bibliography documentation and theoretical studies, and in a practical part which is composed for the design-analysis and the experimental part. This is the dedication in every part expressed in percentage:

- Bibliography documentation 20%
- Theoretical studies 20%
- Design-Analysis 30%
- Experimental part 30%

Also, it will be important to create a planning work. These are the steps to be followed:

- Firstly, it has to be started with the theoretical part. Some basic concepts will be studied and explained. In this way, it will be easier to understand what is doing with NX and interpret and compare the final results.

- Secondly, it will come the NX-SIEMENS research. In this step, the purpose is to learn and get familiarized with the necessary tools of NX by finding information from resources like websites, tutorials and by just trying and testing it. Some patience and time will be required.

- And finally, once this project will be finished, the results will be compared with the objectives in order to get some final conclusions.

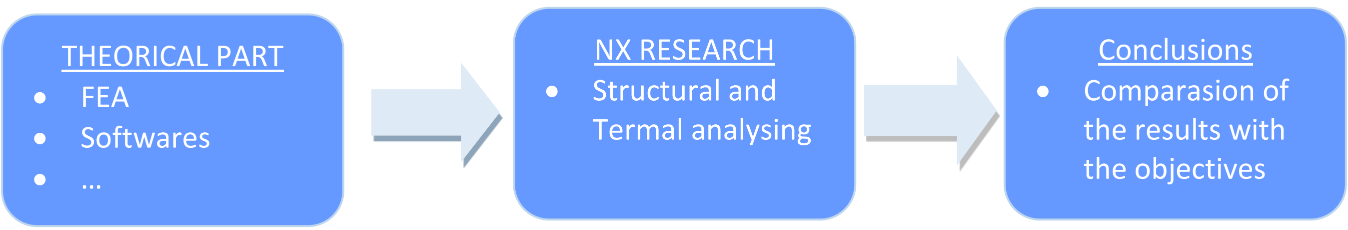


Figure 1. Project guidelines

2 THEORICAL BACKGROUNDS AND CONCEPTS

Before starting with the research part, it will be necessary to have some concepts clear in order to understand what it going to be done. This section will incorporate the main theoretical part of the project and it is divided into two parts:

- Basic concepts: it will be explained the principal idea of the finite elements method and the description of some of the main physical values.

- NX-Siemens: It will be written some basic information about NX-Siemens and ANSYS to comprehend what are these software used for to start getting familiarized with both of them.

2.1 Basic concepts

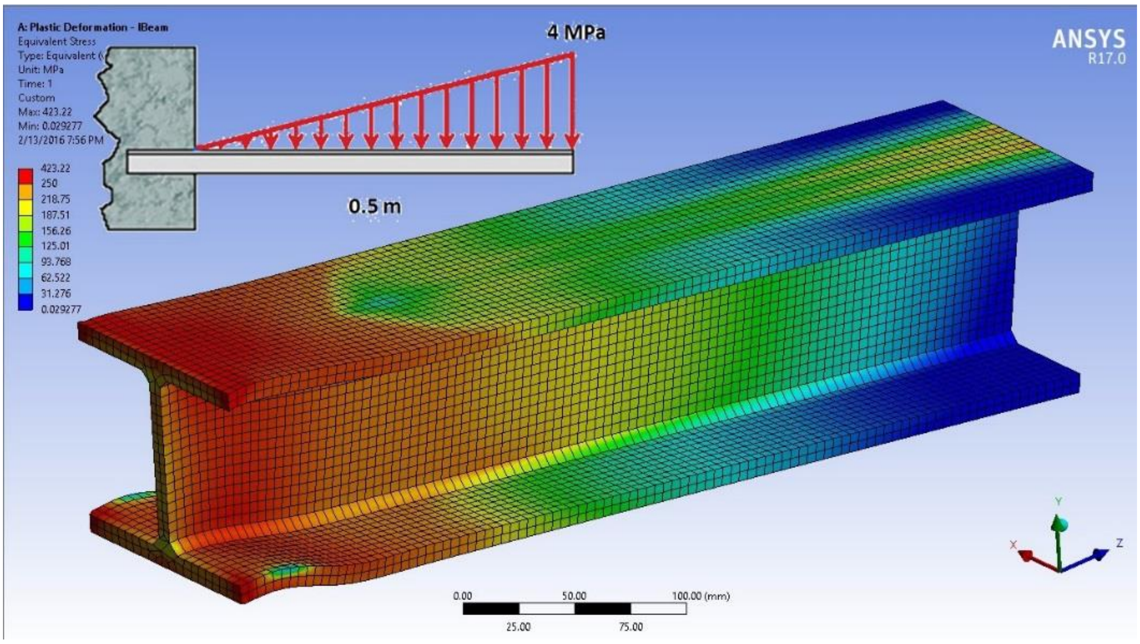
Finite elements method (FEM)

The theoretical parts stars with a simple explanation about FEM as is the main idea from the thesis and the method NX is using for the engineering analysis.

During the studies in the university, students are learnt to solve many different basic problems, all of them which they specific method in order to find an exact solution. But, in the future, it is seen that problems become more and more complex that the ones from the courses. Then, it is realized that the best way to manage these situations is simplifying the problems in order to find approximately solutions. This is the basic concept of FEM.

So, basically, FEM is a numeric method for solving complex problems on the way to find approximately solutions. It is not going to be explained in fully details as it is so much complicated as it seems but, what it is have to be understood is that the way of solving a problem with FEM is diving it in so many small parts so every of these parts become solvable.

In Picture 1, it can be seen a typical representation of an analyzing problem solved with FEM. In this case, it is shown a structural analyzing about the internal stress of a beam which supports a distributed load and fixed from one of its faces to a wall. The beam is divided with a colorful mesh. Every node has a different color, from the range that goes from red to blue, and represents one single approximated solution for the whole area of it. The value in each area is an approximated solution as, actually, we could divide every node in more parts and we would get more and more solutions. The point of FEM it will be choosing how accurate it is want to be.



Picture 1. Structural analyzing with ANSYS of a beam

FEM covers a large different kind of problems. The most common ones are:

- Structural analysis: determination of the effects of loads (forces, deformations or accelerations) applied to a structure or its components.

- Thermal analysis: exchange of thermal energy between physical systems.

- Fluid dynamics: describes the volume flow rate of an fluid (liquid or gas)

<u>Physical values:</u>

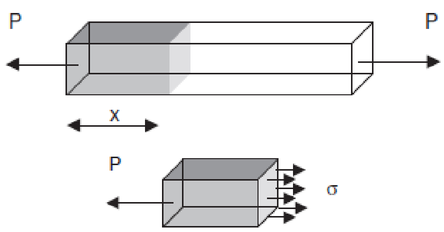
Stress:

Stress is a measure of the internal forces in a body between its particles. These internal forces are a reaction to the external forces applied on the body. In a structural analysis, we will see how our structure react from these external forces and the magnitude and the distribution of the internal forces. To understand the reason of this, it will be important to know how the structures react against any kind of stress.

Stress is classified in these different types:

- Tension:

Is present when two equal but opposite direction forces act on an element and tend to lengthen the material. To have only traction, the effort is located in the center of gravity of the section.



Picture 2. Beam subjected to tension stress

The tensions are studied in the direction of the cut. If the section is cut perpendicular to the effort at a distance x and separate it from the rest, the stress P will give us tensions σ. It is supposed that the tensions are uniform, that means, equal in all the points of the section:

$$\int \sigma \cdot \partial A = \sigma \cdot A = P$$

$$\boxed{\sigma = \frac{P}{A}}$$

The sign of the tension is considered positive as a collective agreement.

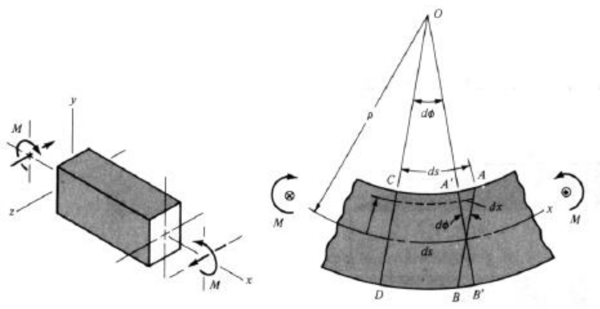
- Compression:

It is present when two equal but opposite forces act on a piece and tend to shorten the material. It is supposed the same hypotheses and identical development with tension, except by the agreement of signs that assigns negative value to the compression

$$\boxed{\sigma = \frac{P}{A}(-)}$$

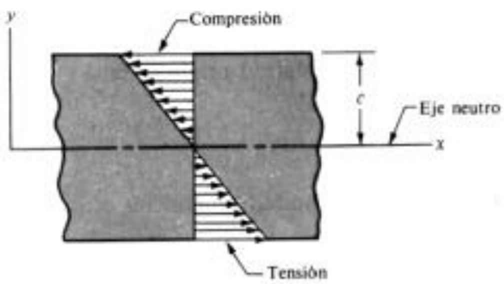
- Bending:

It is the consequence of external forces or moments that produce in the cut section only a bending moment.



Picture 3. Beam subjected to pure bending stress

In the previous figure a beam is shown on which a positive bending moment M acts. The axis Y is the axis of symmetry of the beam. The axis X coincides with the neutral fiber of the beam, and the XZ plane containing the neutral axes of all sections (parallel to the Z axis) is called the neutral surface. The elements of the beam that are on the mentioned surface have zero deformation.



Picture 4. Stress distribution on bending

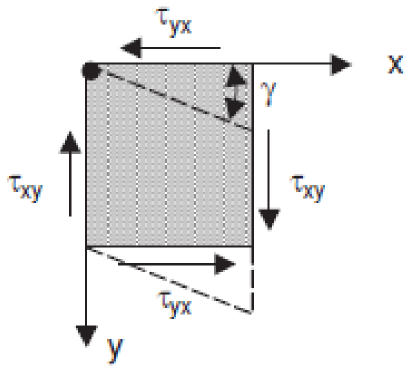
The pure bending equation for the stress establishes that it is directly proportional to the distance from the neutral axis and the bending moment M:

$$\boxed{\sigma = -\frac{M \cdot y}{l}}$$

- Shear:

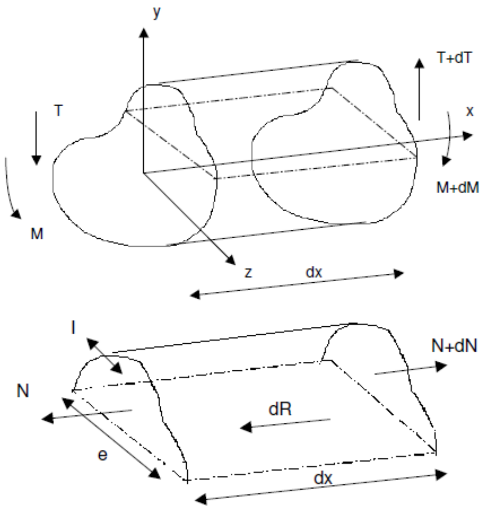
Shear loads provoke the occurrence of shear stresses in the section in which they act. Cutting stresses are characterized because:

- They do not cause volume change, only produce an angular deformation.
- Are equal two to two and converge on the same point



Picture 5. Plane of a beam subjected to shear stress

Most beams have shear forces and bending moments. Only occasionally it is found beams subjected to pure flexion.



Picture 7. Element of a beam subjected to shear stress

The equation of the shear stress is represented by:

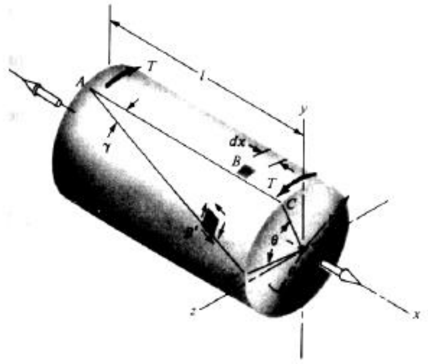
$$\boxed{\tau_{xy} = \frac{T \cdot S_I}{e \cdot I_z}}$$

Where:

- T: Shear force
- I_z: moment of inertia of the whole section
- S_I: static moment of area I

- Torsion:

Any collinear vector with a geometric axis of a mechanical element is called a torque. Let's consider the following recessed cylinder subjected to a torque:



Picture 8. Circular bar subjected to a torque

The torsion stress will be directly related with the angular deformation γ caused by the torque and the shear modulus G so that:

$$\tau = G \cdot \gamma$$

If it is assumed a linear elastic regime, with some steps it can be arrived to the final torsion expression as:

$$\boxed{\tau = \frac{T \cdot \rho}{I_p}}$$

Where:

- I_p: inertia polar moment.
- ρ: outer radious

Heat:

Heat transfer is the physical act of thermal energy being exchanged between two systems by dissipating heat. Temperature and the flow of heat are the basic principles of heat transfer. The amount of thermal energy available is determined by the temperature, and the heat flow represents movement of thermal energy.

In thermal analysis, we will see how our material react from these heat transfers and how its temperature change because of it. As with stress, there are different types of transfer heat. It is generally accepted that heat is transmitted in three different ways named conduction, convection and radiation. These three forms of transfer are the simple ones which can be considered in isolation, although in practice, the normal is that occur, at least, two of them simultaneously.

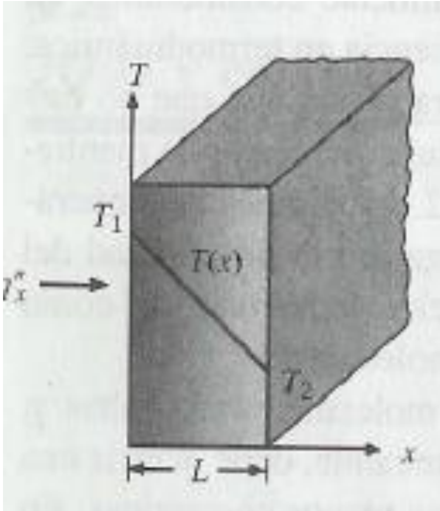
- Conduction:

Conduction is basically a mechanism of energy transfer between contiguous particles. The energy of the molecules increases as the temperature augment. This energy can pass from one molecule to another contiguous one and from this to the next and so on either by shock between particles with fluids or by reticular vibrations with solids.

It is possible to quantify the quantity of heat transfer in terms of appropriate equations or models. These equations or models are used to calculate the amount of energy that is transferred per unit of time. For heat conduction, the equation or model is known as the Fourier's Law represented as:

For the flat wall shown in picture 9, which has a temperature distribution T(x), the one-dimensional equation or model is expressed as:

$$\boxed{q = -k\frac{dT}{dx}}$$



Picture 9. Unidimensional heat transfer by conduction

The heat flow or heat transfer per unit area q (W/m2) is the velocity which the heat is transferred in the direction x per unit area perpendicular to the direction of the transfer, and is proportional to the gradient temperature.

The constant of proportionality k, is a property known as thermal conductivity (W/m·K) and is a characteristic of the material of the wall. The minus sign is a consequence of the fact that the heat is transferred in the direction of the decreasing temperature.

- Convection:

In contrast to conduction, convection involves transport of energy and matter, therefore, this form of heat transfer is possible only in fluids.

Defining it as simple as possible, convection is the mechanism of heat transfer produced by movement of mass or circulation within the substance. It can be natural produced only because of the differences densities of the matter or forced when matter is forced to move from one place to another.

The general equation for convection is called Newton's law of cooling and is represented as:

$$\boxed{q = h \cdot A \cdot (T - T_{\text{env}})}$$

Where:

- q: is the thermal energy in joules
- h: is the heat transfer coefficient (W/m^2K)
- A: is the heat transfer surface area (m^2)
- T: is the temperature of the object's surface and interior.
- T_{env}: is the temperature of the environment.

- Radiation:

Thermal radiation is the energy emitted by matter with a finite temperature. In thermal radiation, heat is transmitted by electromagnetic waves, like light, but with different wave lengths. The radiant energy depends on the characteristics of the surface and the temperature of the emitting body. When it hits a receiver, part of the energy passes to this other body, depending on the characteristics of the same and its power of absorption. This energy makes increase the temperature of the second body. The transfer of heat by radiation only involves the transport of energy.

The amount of energy leaving a surface in the form of radiant heat depends on the absolute temperature and the nature of the surface. A perfect radiator or blackbody emits an amount of radiant energy from its surface per unit of time by the following equation:

$$\boxed{q = \sigma \cdot A \cdot T^4}$$

Where σ is the Stefan Boltzmann constant $\sigma = 5.67 \cdot 10^{-8}\ W/m^2K^4$

3.2 NX-Siemens

NX-SIEMENS (NX)

NX is an advanced high-end software package that provides the following solutions:

- CAD (Computer-aided): this software makes product design more efficient by automating processes that were once manual, such as traditional drafting. You can also do solid modeling (creating 3d representations of your designs)
- CAM (Computer-aided manufacturing): it is used to control manufacturing machine tools like lathe, milling or electrical discharge machine.
- CAE (Computer-aided engineering): is the software related with the engineering analysis simulations.

The thesis is going to be focused on the CAE software part as is the one related with the simulations. The simulations are going to be tested are structural and thermal analysis:

Structural analysis:

Understanding how a component or product assembly reacts under stress or vibration is critical in any industry, but as products and materials become increasingly complex, engineers need tools that go beyond linear statics and dynamics analyses. NX is one of these tools. Some problems that it can solved are about linear analysis, non-linear analysis, structural dynamics, durability and fatigue analysis and noise, vibration and harshness (NVH) analysis.

Thermal analysis:

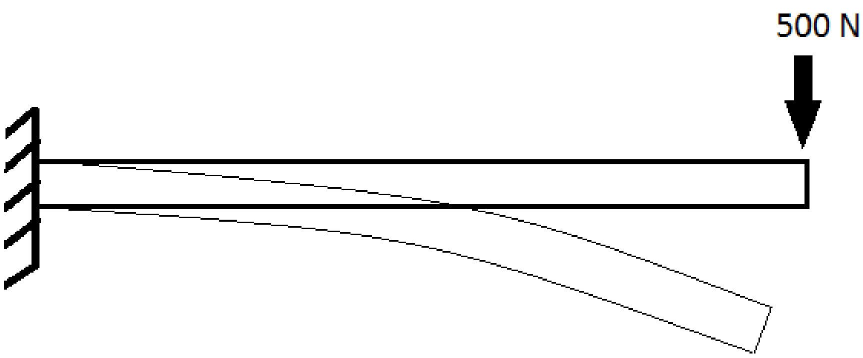
Thermal management is a major consideration for a wide range of products, including industrial machinery, automobiles and consumer electronics. The objective of any thermal management solution is to maintain a product’s temperature within a range that is optimal for performance. Accomplishing this may require the removal or addition of heat, either passively or in an actively managed fashion, and this can be evaluated using thermal analysis software. In NX it is going to be able to find solutions with thermal problems related in the three type of heat transfer (conduction, convection and radiation).

4 RESEARCH PROJECT

Now that it has been seen all the important concepts, it is time to start with the research part. As it is been mentioned, the purpose in this area is explaining how to solve structural and thermal analysis in NX. An easy instruction format is going to be written and compared what is explained with a practice example. As the software is so complex, it is not able to study it so deeply so, in some way, the project will be used to have a first impression and knowing some basic tools.

Structural analyzing:

Before starting, it is need a problem which is want to be solved using a structural analyzing. As starters with the software, it has been planned an easy static mechanic problem that will consist of a beam with a fixed restriction in one of its faces and a force on the other one. The problem is represented in the following drawing:



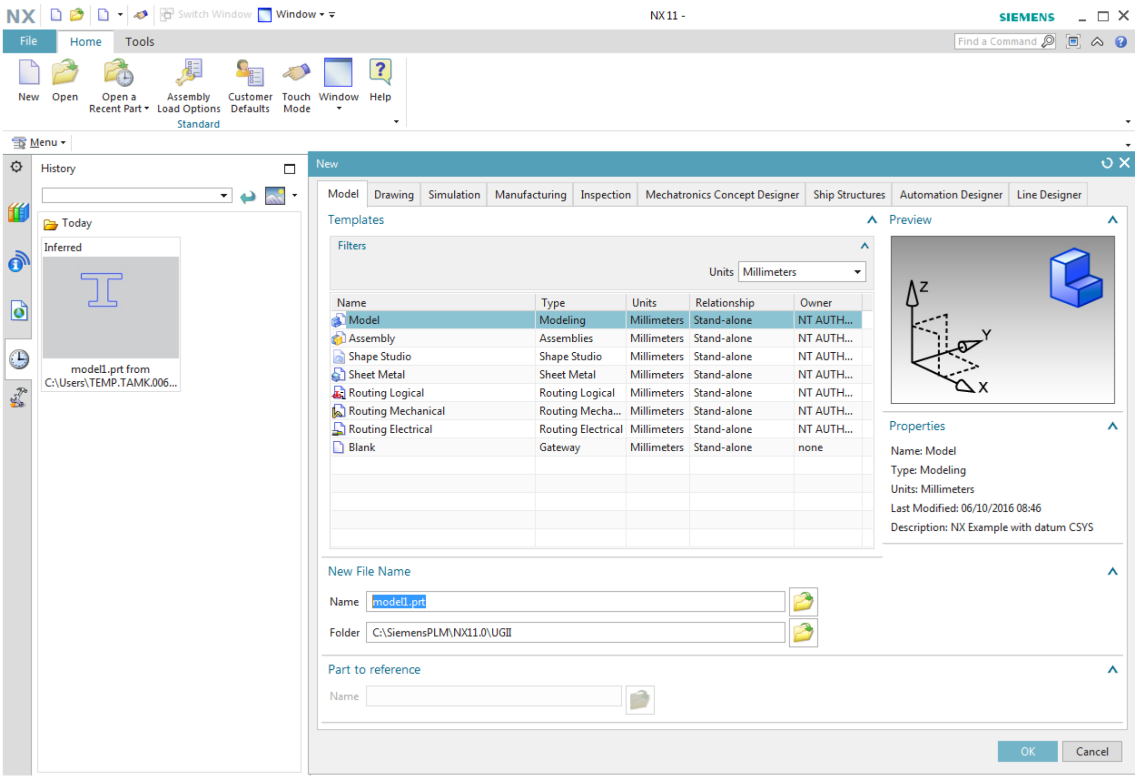
Picture 10. Structural analyzing problem

The objective, first of all, is to define this problem in NX to then analyze it to see how the beam will react internally over this situation. The three main points to be seen are:

- The internal stress.
- The deformation of the material.
- The reaction forces.

Let´s start to solve the problem with NX. First of all, NX needs to be opened. It is used the version NX 11 as is the license TAMK has actually.

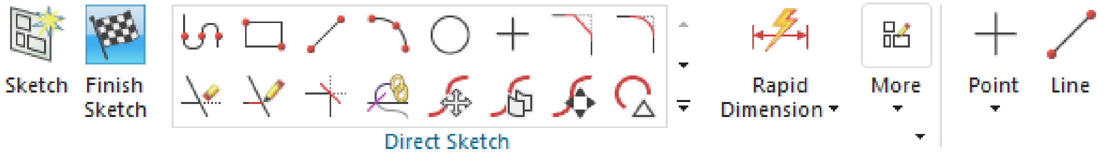
Picture 11 is the home screen from NX11. There, "new" has to be selected to open the previous window and there it is has to be written the name it is wanted for the file and where it is wanted to save it. After that it is just need to click on "OK".



Picture 11. Home screen from NX11

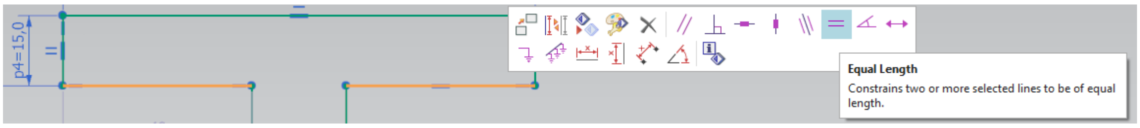
To start to work with the analysis it is need to model the structure. As it is said, the structure will consist on a beam as is easier and more realistic to see. There are a lot of ways to modeling it. It is going to be taken the easy way creating a simply sketch of the section of the beam and extracting it. Another way to create the beam would be importing steps file from parts created by other software as Solidworks…

To start with the sketching, it is just need to click on sketch (picture 12) and then select a plane.



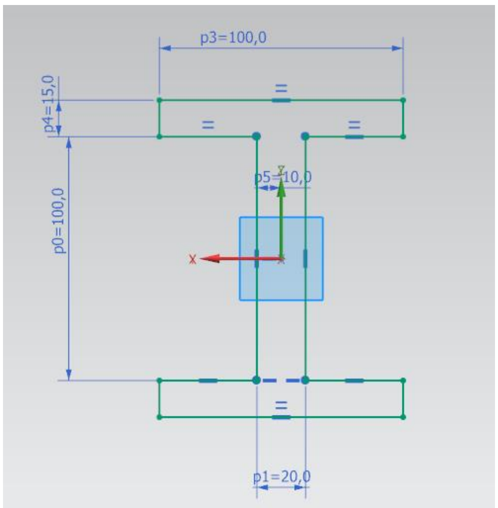
Picture 12. Basic tools for sketching

After that, a draw of the section has to be done with the line option (picture 12). With rapid dimension is going to be easy to have our section finished. Also, if it is wanted to make it simpler and more clear, it can be use restrictions as making lines collinear, parallel or equal length (picture 13) just clicking two or more lines, points…



Picture 13. Main restriction

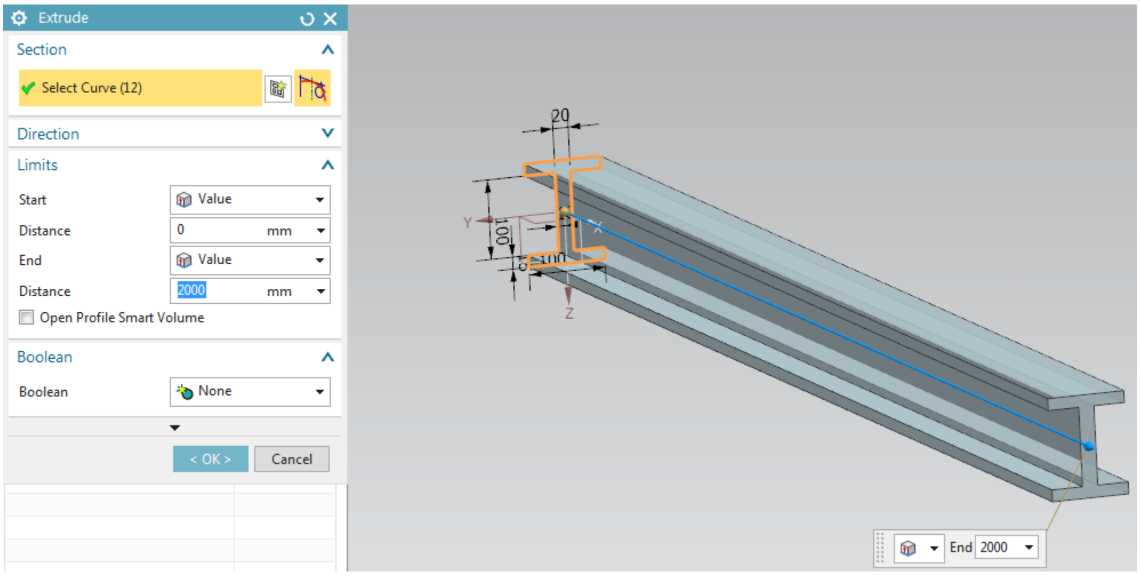
These are going to be the dimensions for the beam section (picture 14).



Picture 14. Sketch of the section from our beam

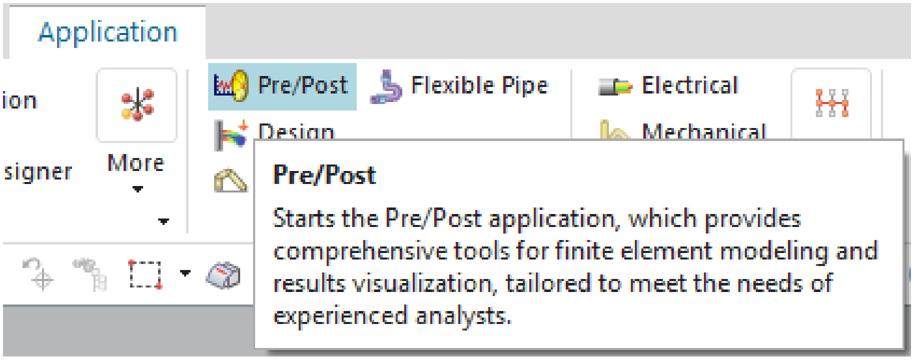
It is only need to make the extrusion. For that, there is an easy option called extrude. 

After clicking it, it is set the distance of the extrusion. In this case, it is selected 2000mm (Picture 15).



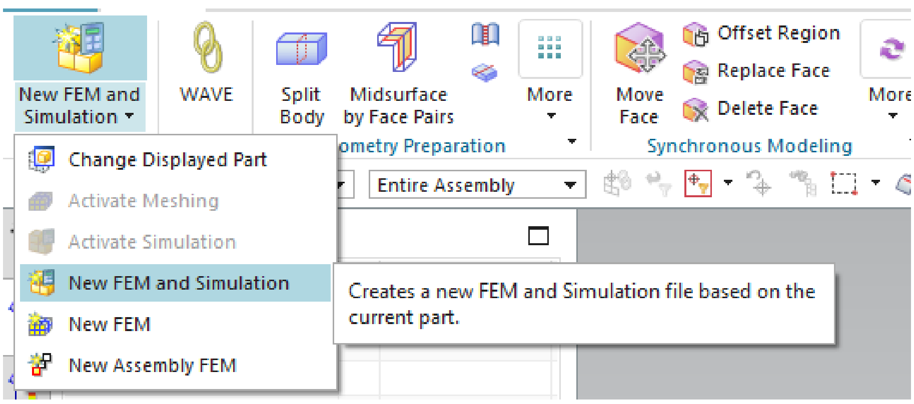
Picture 15. Extrusion mode

Now the beam is modeled and it is going to be time to start with the structural analysis going on Application and clicking on Pre/Post (Picture 15).



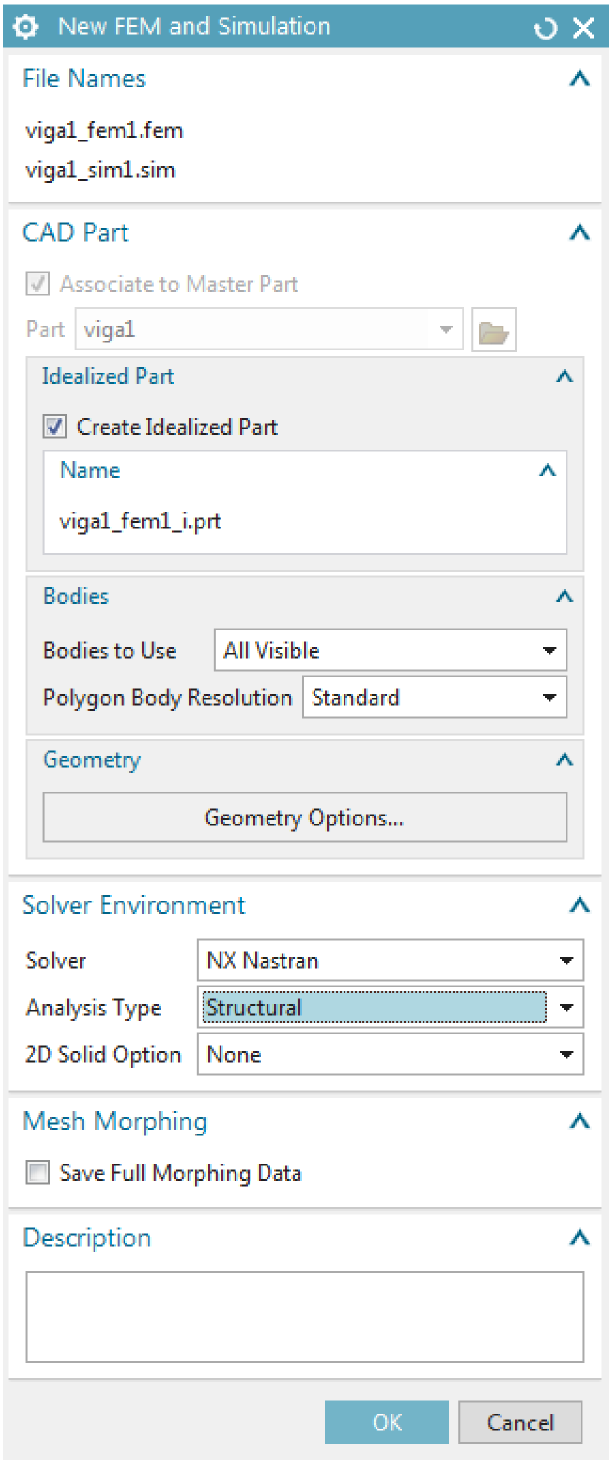
Picture 16. Pre/post tool

In the new window is going to be opened, let's click directly to new FEM and simulation. (Picture 16)



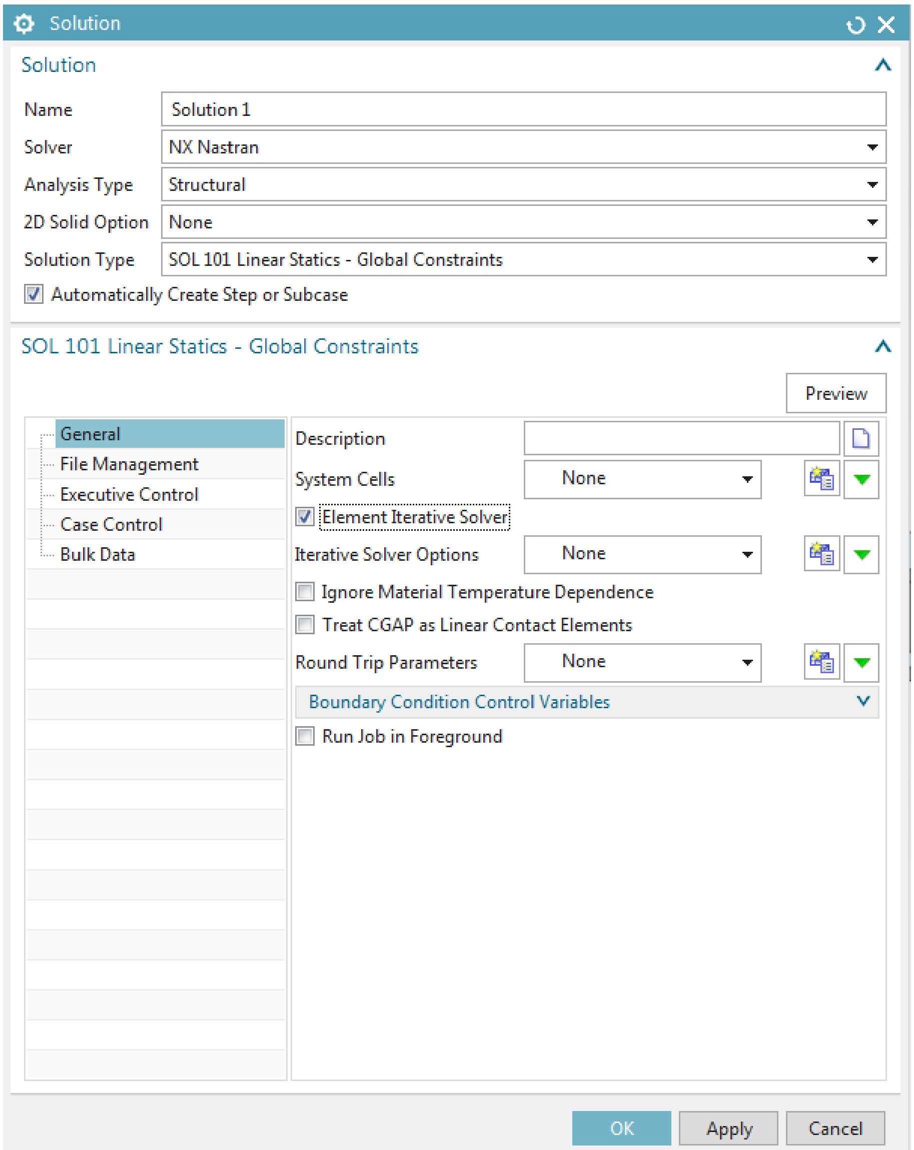
Picture 17. New FEM and Simulation bar

Then, it has to make sure all the parameters are these ones (picture 18). The most important to check out on these parameters is that it is using NX Nastran as is the solver for stress, vibration, buckling, structural failure, heat transfer, acoustics and aero elasticity by NX Siemens using FEA. And also, it has to be chosen structural as analysis type to have all the tools necessary for the example.



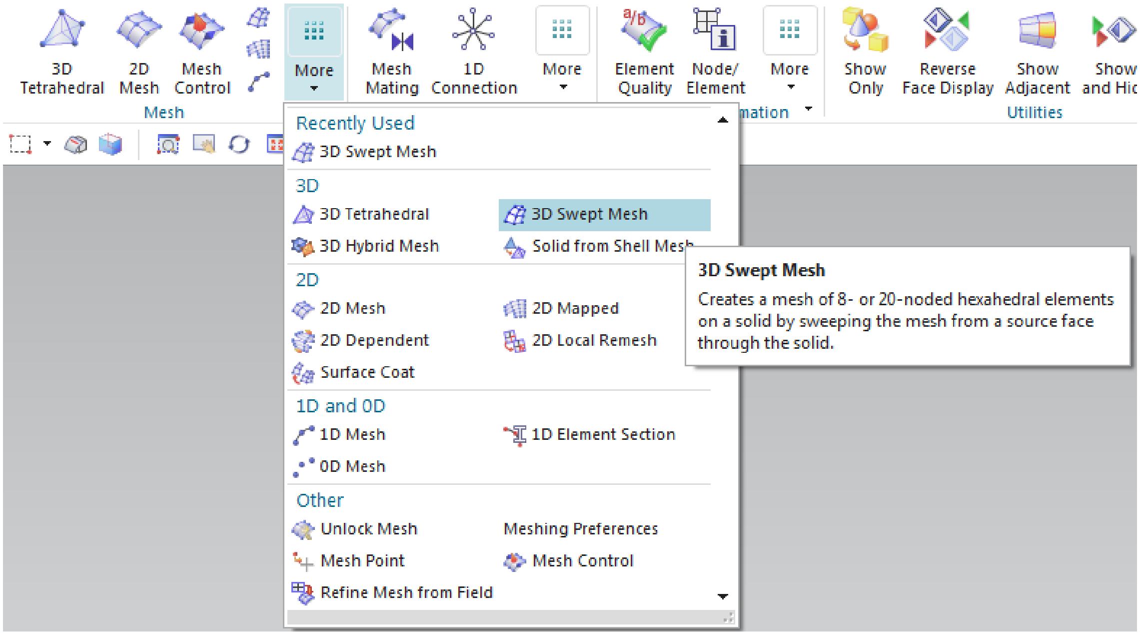
Picture 18. FEM and simulation parameters

From here, the following window will be directly opened where the type of solution will be chosen. In this case, it is chosen linear statics solution. It could be chosen a non-linear solution if it is given non-linear properties to the material but it is not this case as, then, it gets so much difficult and that is not the purpose. Also, it is recommended to activate the Element Iterative Solver option (Picture 18) when working with solids.



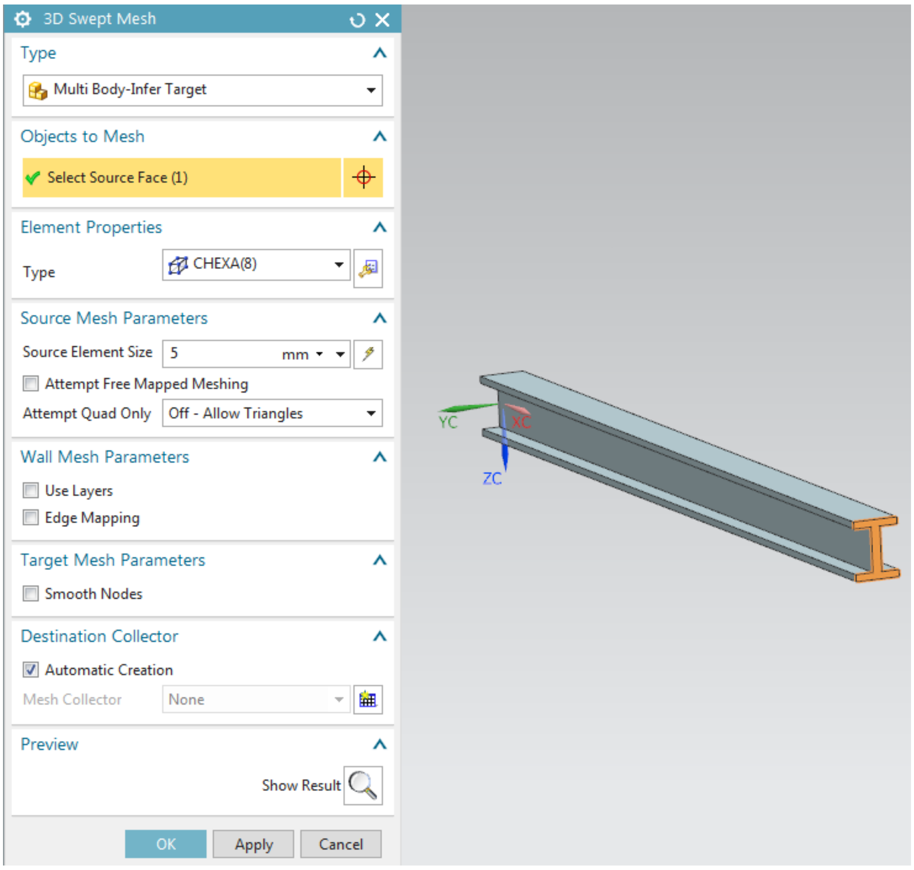
Picture 19. Solution window

Now it is time to create a mesh for the beam. There are a lot of options for that as it can be seen in the toolbar. The option that fits more with the beam is a 3D Swept Mesh (Picture 20).



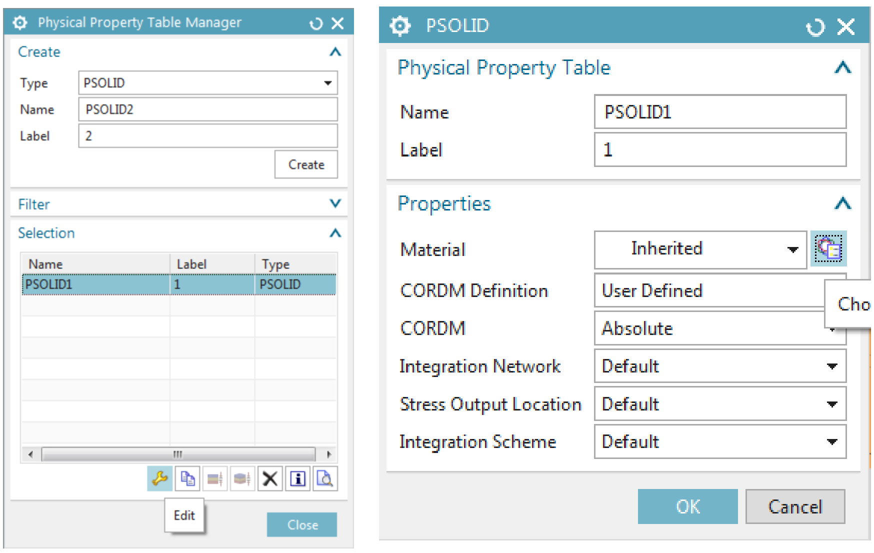
Picture 20. Mesh toolbar

After clicking there, it is going to be chosen the type of mesh it is wanted where CHEXA 8 (8 node mesh) is totally enough for the problem. Next, it is going to be selected the face of the beam and automatically NX will recognize since where it is wanted to create the mesh. To end with this part, it is necessary to pick the size of the mesh that will be 5mm, as in this way, it is going to be created 3 exactly node in the upper part as it measures exactly 15mm and it will lock so much clear.



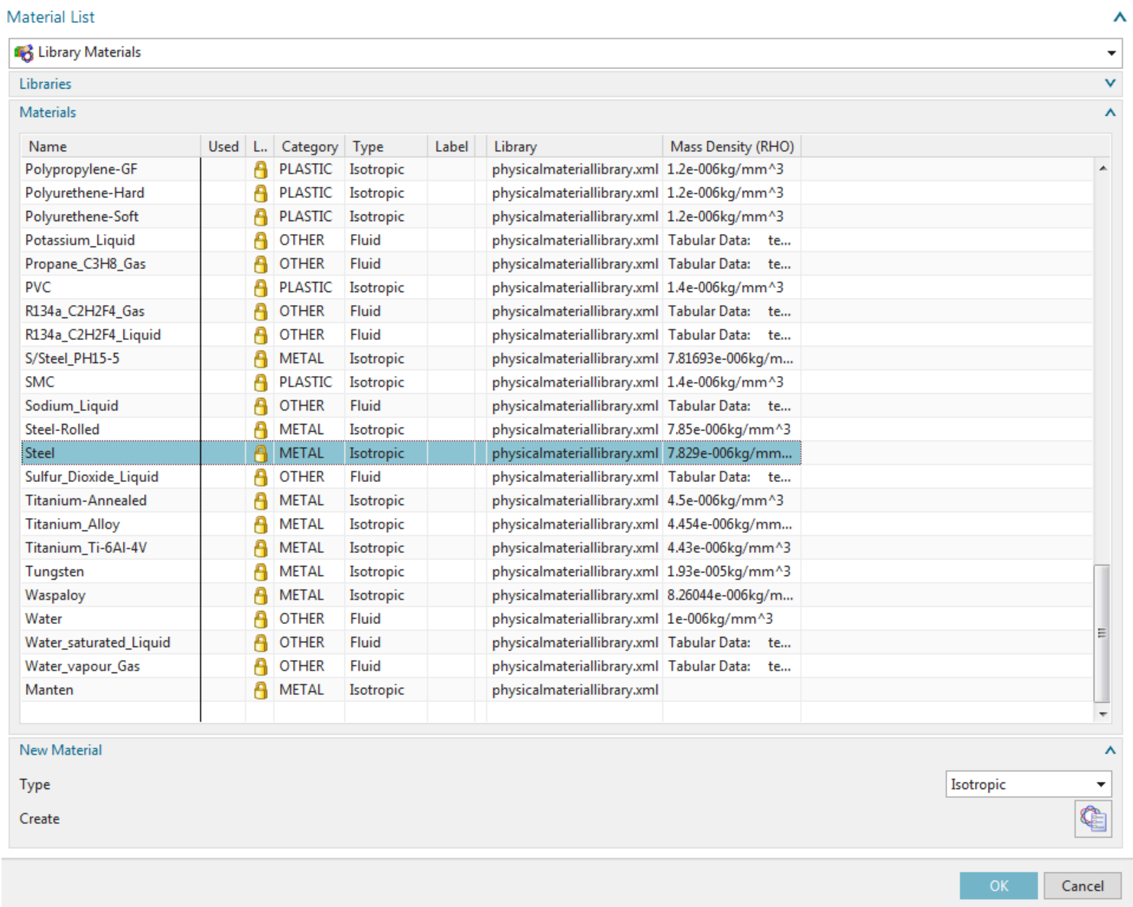
Picture 21. 3D Swept Mesh

Before starting with the restrictions of the structure, there is one more important step that it cannot be forgotten which is setting a material for the beam. To go with this action, it must be clicked material property, where solid has to be selected, then edit and after library materials ha to be opened (picture 22).



Picture 22. Physical property table manager

The library materials contain a quite large list of different material alphabetically ordered and classified for his category (metal, plastic or other), his type (isotropic or fluid) and his mass density. For this example, it is going to be used steel material. Then just have to click ok with the default parameters and the material will be finally defined.

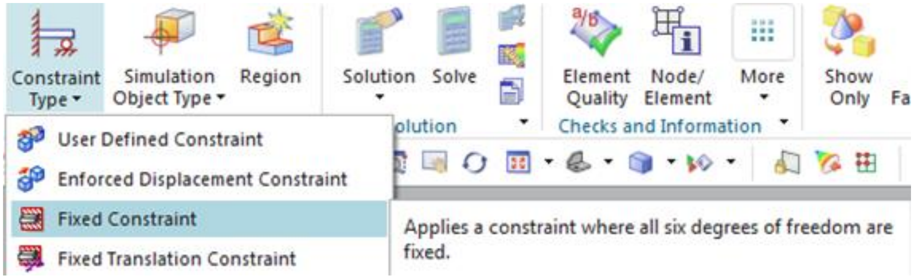


Name	Used	L..	Category	Type	Label	Library	Mass Density (RHO)
Polypropylene-GF			PLASTIC	Isotropic		physicalmateriallibrary.xml	1.2e-006kg/mm^3
Polyurethene-Hard			PLASTIC	Isotropic		physicalmateriallibrary.xml	1.2e-006kg/mm^3
Polyurethene-Soft			PLASTIC	Isotropic		physicalmateriallibrary.xml	1.2e-006kg/mm^3
Potassium_Liquid			OTHER	Fluid		physicalmateriallibrary.xml	Tabular Data: te...
Propane_C3H8_Gas			OTHER	Fluid		physicalmateriallibrary.xml	Tabular Data: te...
PVC			PLASTIC	Isotropic		physicalmateriallibrary.xml	1.4e-006kg/mm^3
R134a_C2H2F4_Gas			OTHER	Fluid		physicalmateriallibrary.xml	Tabular Data: te...
R134a_C2H2F4_Liquid			OTHER	Fluid		physicalmateriallibrary.xml	Tabular Data: te...
S/Steel_PH15-5			METAL	Isotropic		physicalmateriallibrary.xml	7.81693e-006kg/m...
SMC			PLASTIC	Isotropic		physicalmateriallibrary.xml	1.4e-006kg/mm^3
Sodium_Liquid			OTHER	Fluid		physicalmateriallibrary.xml	Tabular Data: te...
Steel-Rolled			METAL	Isotropic		physicalmateriallibrary.xml	7.85e-006kg/mm^3
Steel			METAL	Isotropic		physicalmateriallibrary.xml	7.829e-006kg/mm...
Sulfur_Dioxide_Liquid			OTHER	Fluid		physicalmateriallibrary.xml	Tabular Data: te...
Titanium-Annealed			METAL	Isotropic		physicalmateriallibrary.xml	4.5e-006kg/mm^3
Titanium_Alloy			METAL	Isotropic		physicalmateriallibrary.xml	4.454e-006kg/mm...
Titanium_Ti-6Al-4V			METAL	Isotropic		physicalmateriallibrary.xml	4.43e-006kg/mm^3
Tungsten			METAL	Isotropic		physicalmateriallibrary.xml	1.93e-005kg/mm^3
Waspaloy			METAL	Isotropic		physicalmateriallibrary.xml	8.26044e-006kg/m...
Water			OTHER	Fluid		physicalmateriallibrary.xml	1e-006kg/mm^3
Water_saturated_Liquid			OTHER	Fluid		physicalmateriallibrary.xml	Tabular Data: te...
Water_vapour_Gas			OTHER	Fluid		physicalmateriallibrary.xml	Tabular Data: te...
Manten			METAL	Isotropic		physicalmateriallibrary.xml	

Picture 23. Library materials

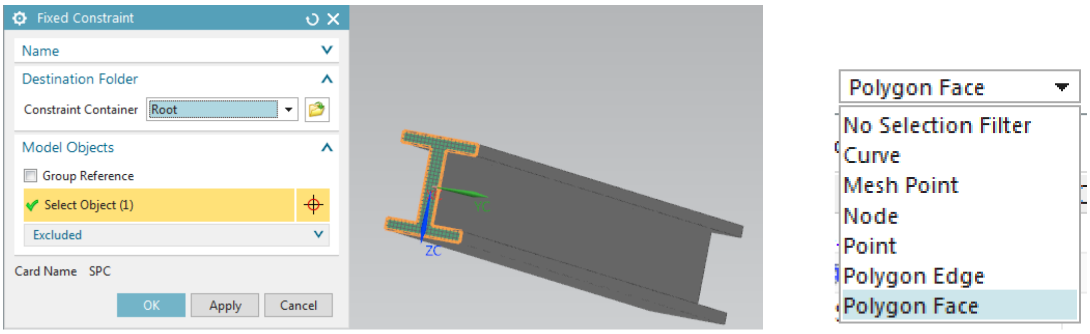
Once the structure is completely modeled and defined, it is time to input the restrictions of the problem are going to analyzed. In a basic and lineal structural analysis, these restriction will be loads and constrains. This software allows a wide range of different restrictions.

Starting with the restrictions, it is seen that the beam is totally fixed from one face (picture 10). For that, it is necessary a fixed constraint from the Constrain type menu from the toolbar (picture 24).



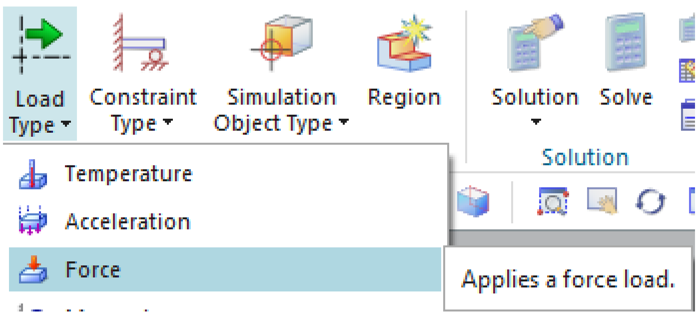
Picture 24. Fixed constraint

To set the fixed constrain it is only necessary to select the face is wanted to fix. It is important to establish polygon face in the way to choose faces from the part (Picture 24).



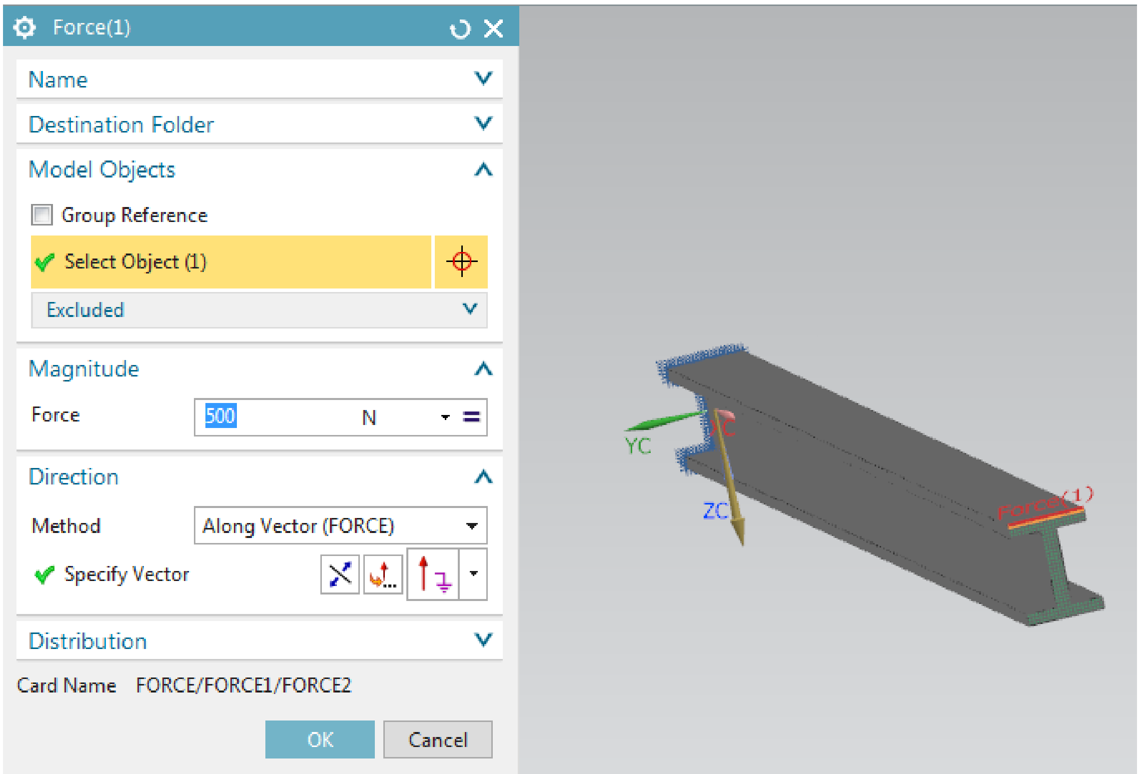
Picture 25. Selecting face for fixed constrain

About the load restrictions, it is need to input just one force on the opposite face. That option is quite easy as well. It will be found all the loads on Load type from the toolbar and then force will be selected.



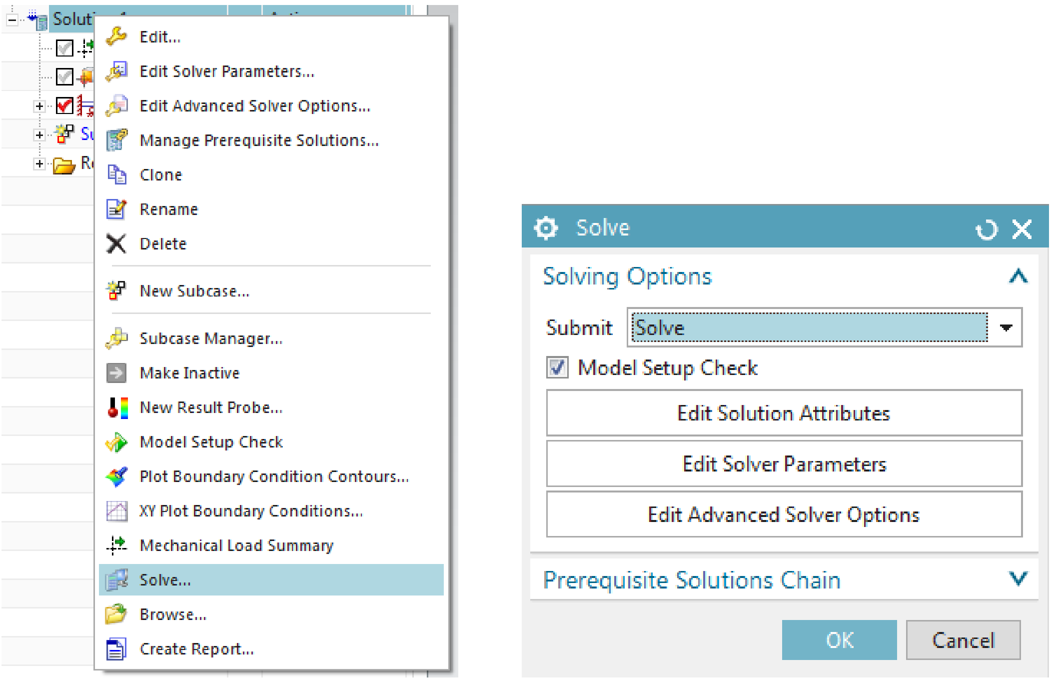
Picture 26. Load Types

As with the constraints, it is need to be selected where the force is. In this case, it is not going to be selected a face but an edge instead. So, it is need to impose on the filter polygon edge. Once there, it must be set the magnitude of force (500N) and the sign of the vector.



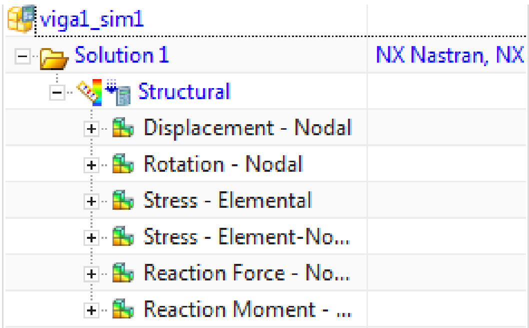
Picture 27. Force parameters

Now, that the problem is set up in NX, it is time to analyze it. This is, maybe, the easiest part as the software will solve all the problem by itself. What is need to be done is right click on solution and then normal click on Solve. The next window will be skipped as it stays with the predetermined solve parameters.



Picture 28. Solve options

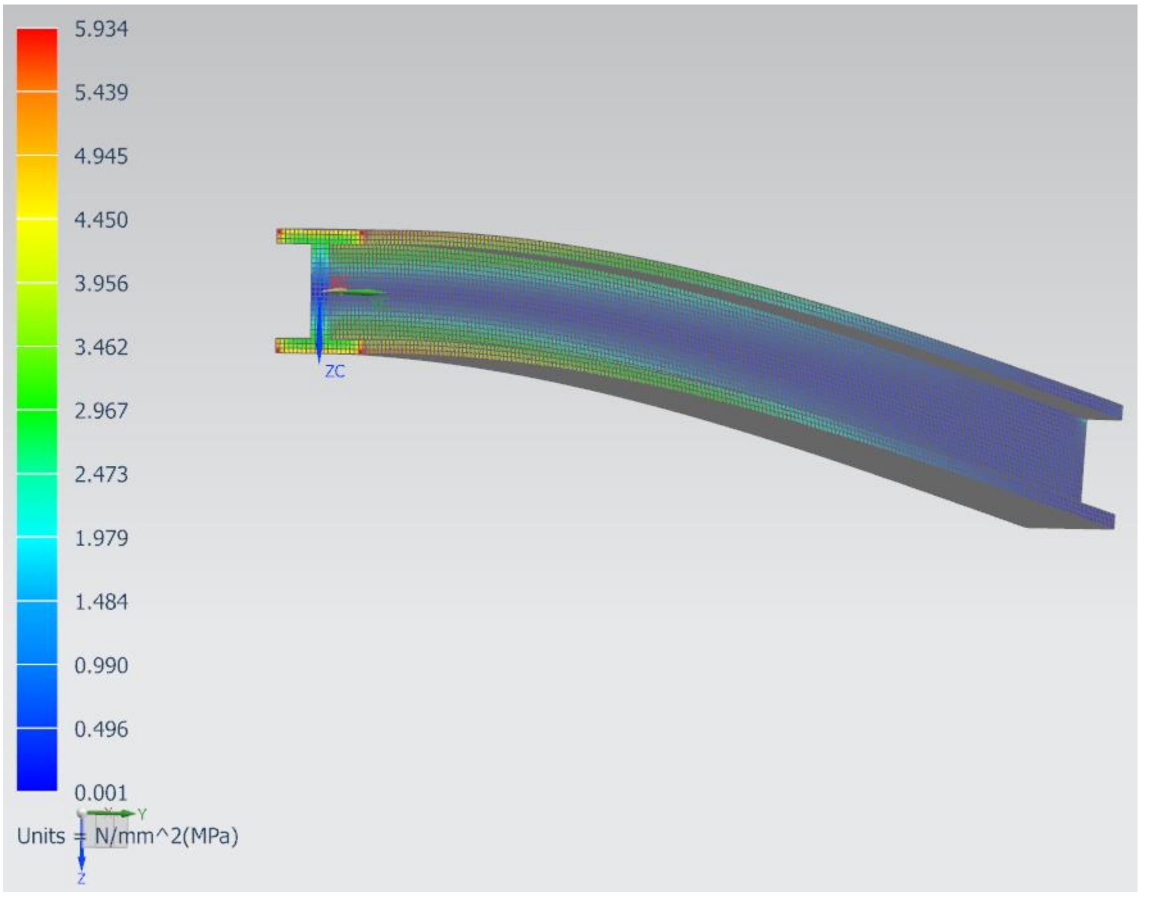
After that, NX will take some seconds to analyze the problem. Some windows will be opened during the process to the end. When in one of them it is said that the problem is resolved, all of them should be closed. A list of solutions can be seen by clicking on solution 1 folder and structural then.



Picture 29. List of options

In the list of solutions, it is going to be focused on the ones mentioned before:

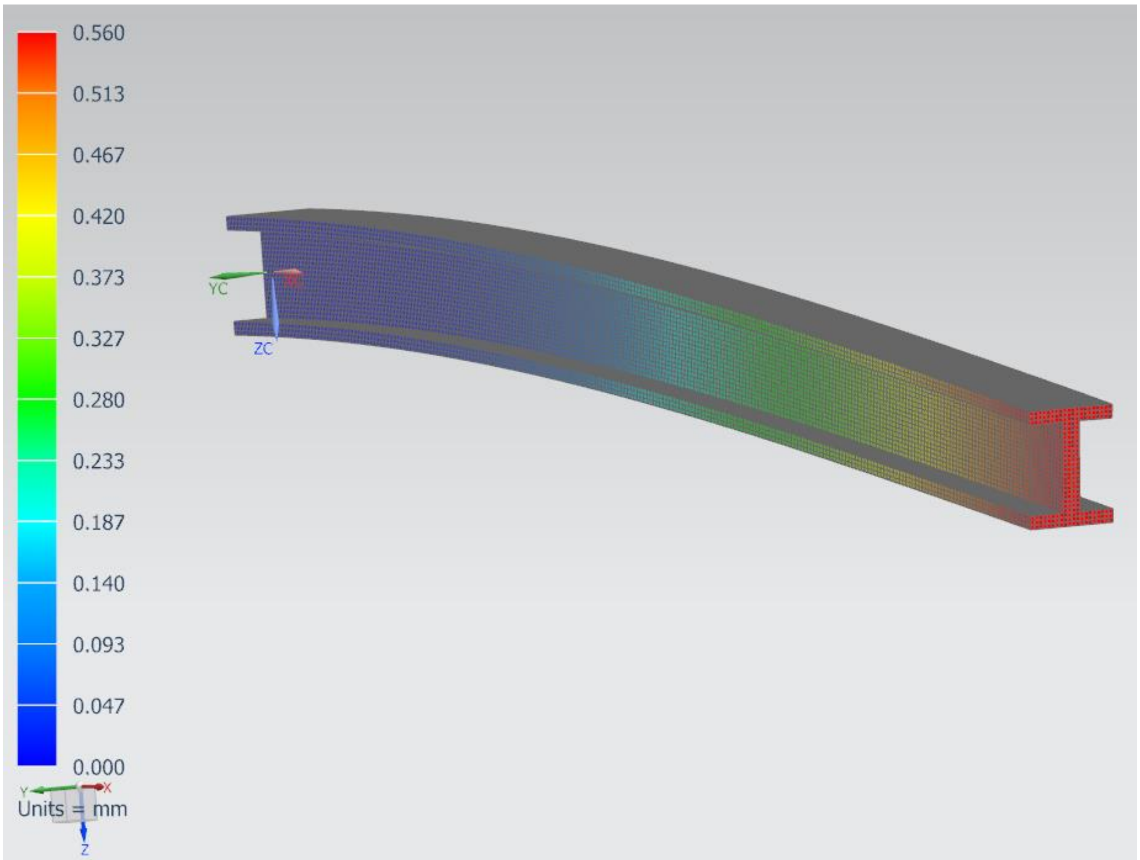
Internal stress:



Picture 30. Internal stress representation

This first solution represents the internal stress of the beam by Von Mises yield criterion (Picture 30). It can be accessed by clicking stress elemental in the list of solutions (picture 29.) The stress is represented by a range of different colors going from blue to red. The redder the color is, the huger the stress is. As the force cause, basically, a flexion stress, this will be distributed far from the neutral axis, as it has been seen in the theoretical part (Picture 4), with a maximum valor of 5,934. Thanks to the colorful representation, it can be seen the critic zone from where the beam will start to rupture in case the stress exceed the ultimate strength.

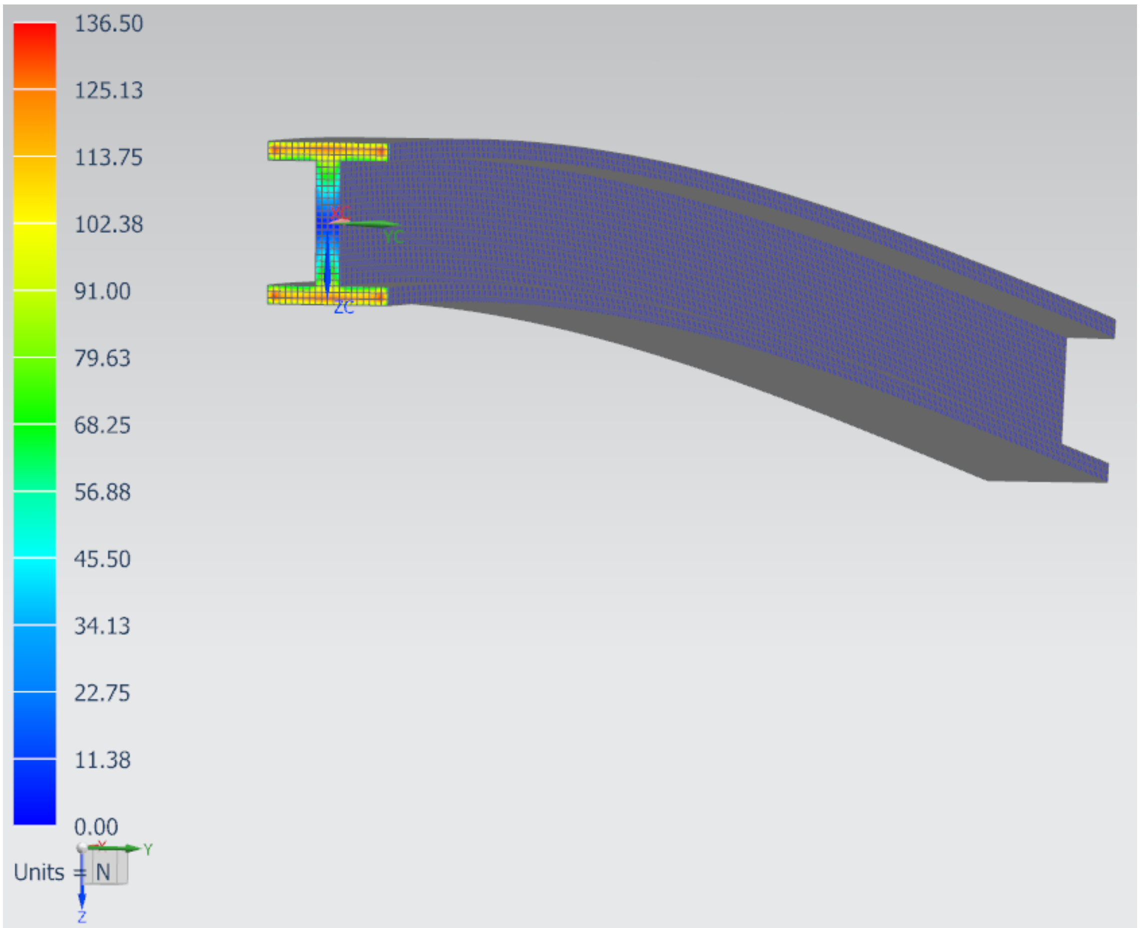
<u>Deformation of the material:</u>



Picture 31. Deformation of the beam

The second solution represents the deformation of the material by mm (Picture 31). Is the one called displacement – nodal from the list of solutions. As the previous solution, the beam is distributed by the same range of color but this time it represents the displacement of the material by mm. The redder is, the huger the displacement. Logically, the maximum deformation take place where the force is been applying with a maximum valor of 0.560mm. Also is represented how the beam would be bended in an exaggerated way as it is seen that the maximum valor does not correspond proportionally. It can be changed the scale to check the deformation in a real scale very easy in the toolbar.

Reaction forces:

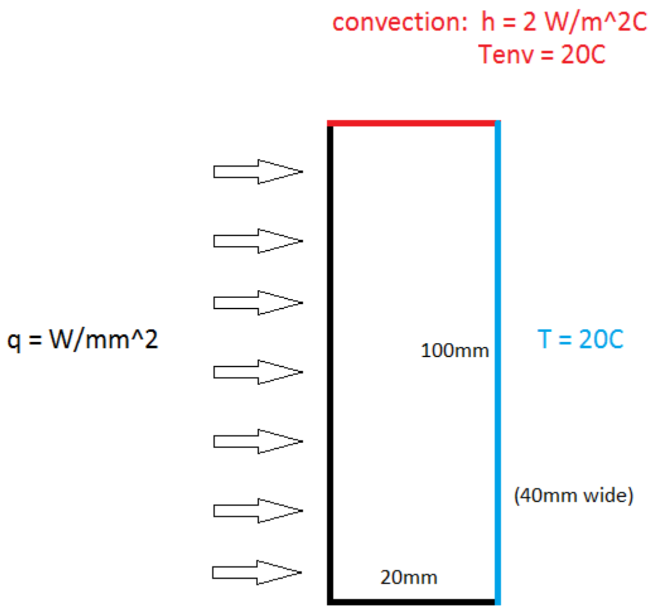


Picture 32. Internal forces

The last solution it is going to be seen is the one that represents the internal forces. It is named reaction force in the list. This time, the range of colors represents the internal forces caused by the load. As it is represented, there are only reaction forces where the beam is fixed as it is where the restriction is. The maximum valor of this force is 136.5N.

<u>Thermal Analysis:</u>

As with the structural analysis, it is need a problem to start with. This is going to be the problem for the thermal analysis which has been chosen properly to be easily to understand:



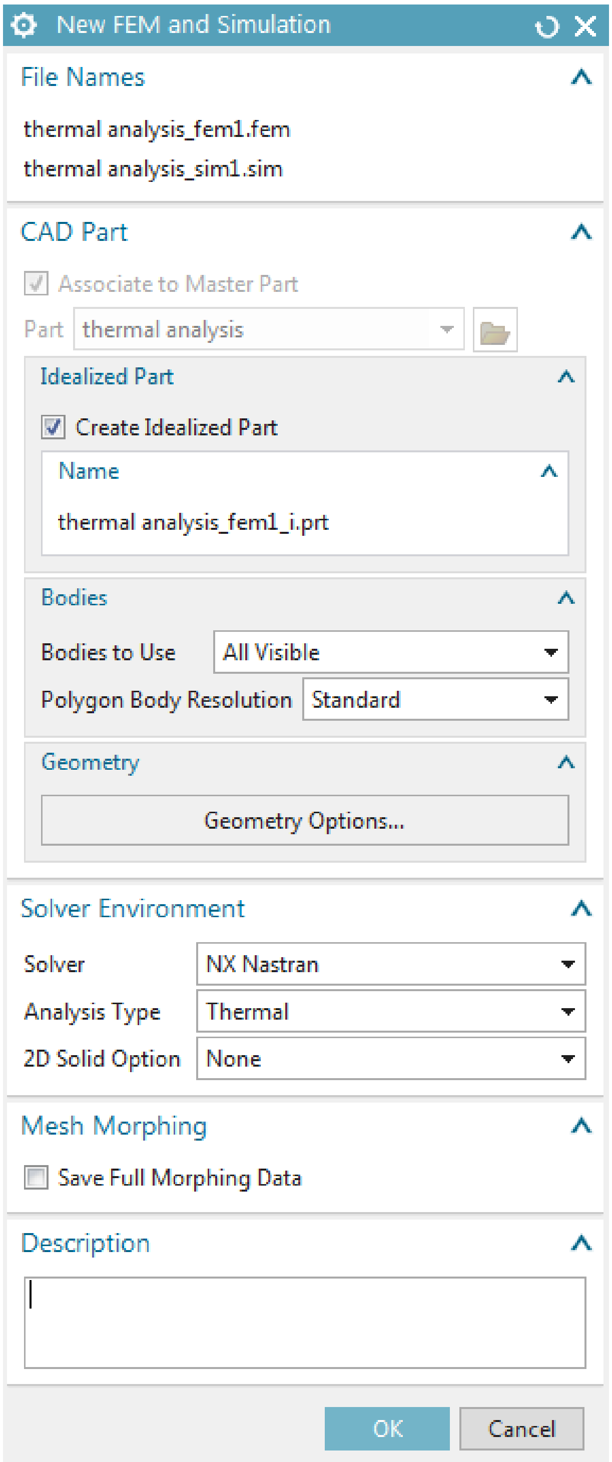
Picture 33. Thermal analysis problem

As it can be seen in the picture 33, there is a wall with the mentioned dimensions. The wall has a constraint temperature of 20º at the end, a 1 W/mm^2 heat uniform flux coming from the opposite side and a convection transmission affecting the top of it.

Before starting with the explanation, there the need to mention that in this example are going to be some steps to do very similar from the first example. So, in order to not being repetitive, they are not to be explained as explicit as before.

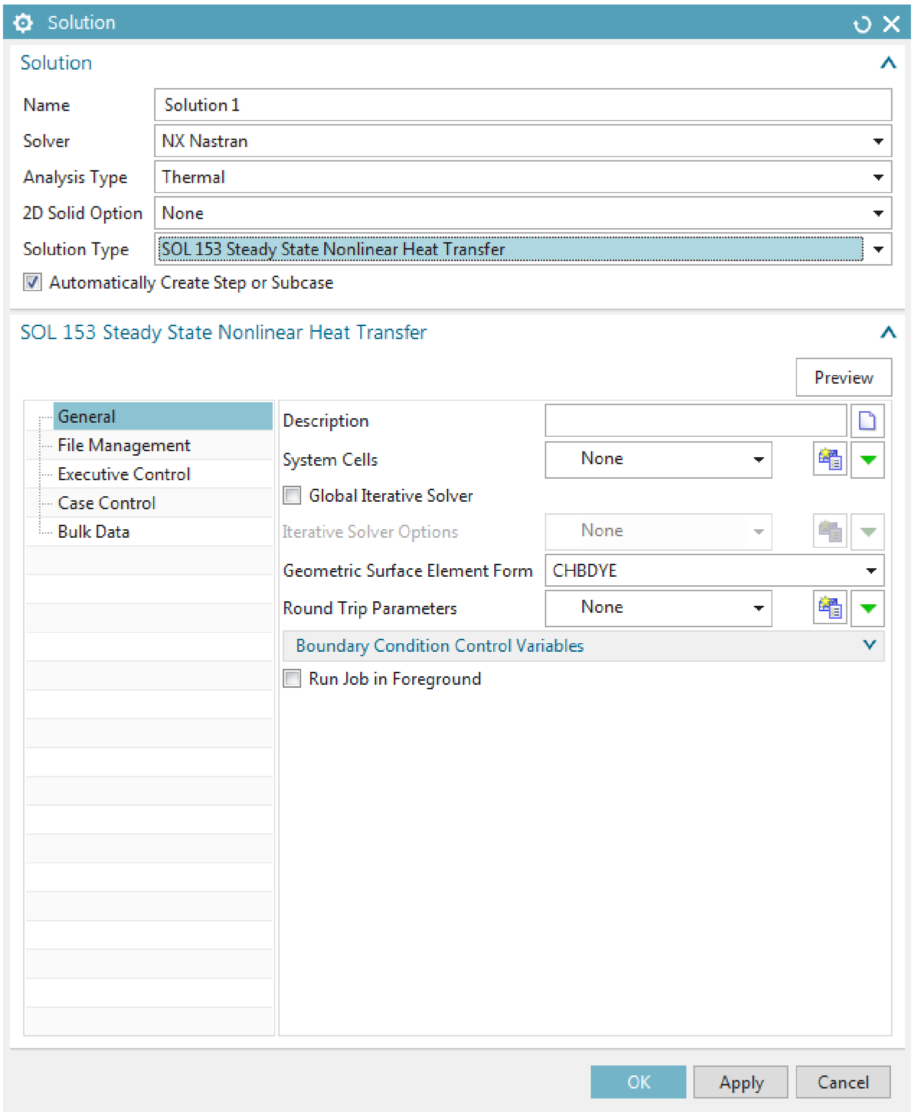
Let's start with the thermal analysis explanation. First of all, it is need to model the wall. For that, a sketch has to be created and extruded. The steps are the same as the previous example.

After wall is modeled with the dimensions from the drawing, it is time to start with the thermal analysis. As it has done with the structural analysis, Pre/post icon has to be clicked to open then the New FEM and Simulation window. This time it is has to be chosen the Thermal Analysis type.



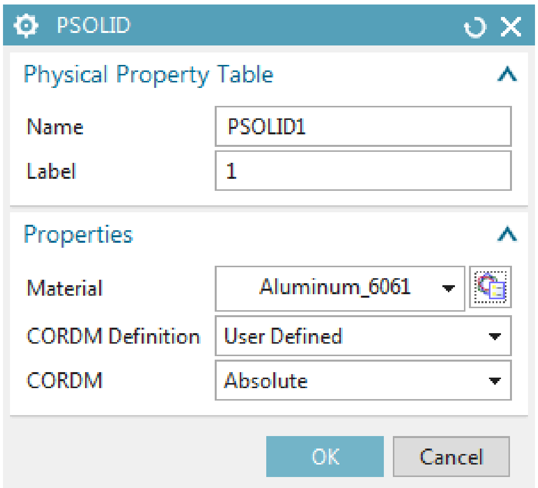
Picture 34. New FEM and Simulation

Consecutively, the Solution window will be opened. There, it can be chosen a steady state or a transient solution type depending on what is going to do. With this problem and in order to make things easier, is it better starting with the steady state option.



Picture 35. Solution set up

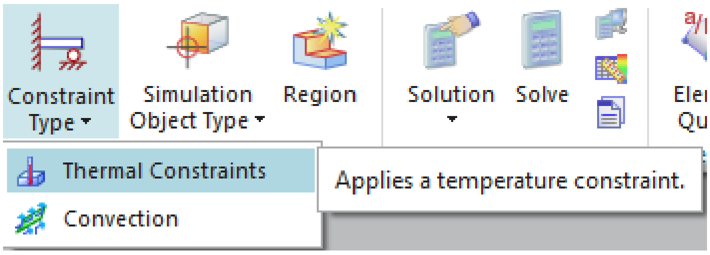
The next 2 steps, which are defining the mesh and the material, are going to be done the same way as with beam from the previous problem. With the mesh, this time, it is suggested to set a source element size of 2.5mm because of the actual dimensions to assure a clear FEM representation. About the material, it is going to use an Aluminium_6061 (Picture 36), where it can be found from the same material list.



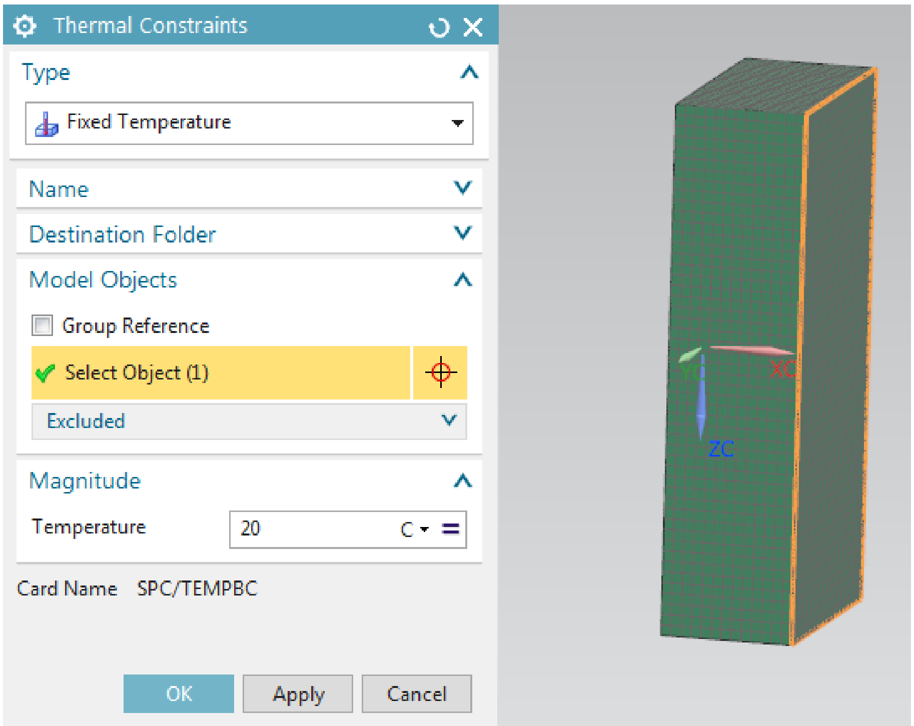
Picture 36. Aluminium_6061 selection table

Now, it's about time to start with setting the restrictions from the actual problem. The process is most exactly the same just constrains and loads are now fluxes and temperatures instead of forces and kinematic constrains.

Starting with constrains, let's click on constrain type from the toolbar (picture 37). There, there are the options of doing a thermal constraint or a convection. First, it is going to be set the temperature at the end of the wall clicking on thermal constraint. Then, it is going to be chosen the face where it is want to define the temperature and the magnitude of it that will be 20ºc (picture 38).

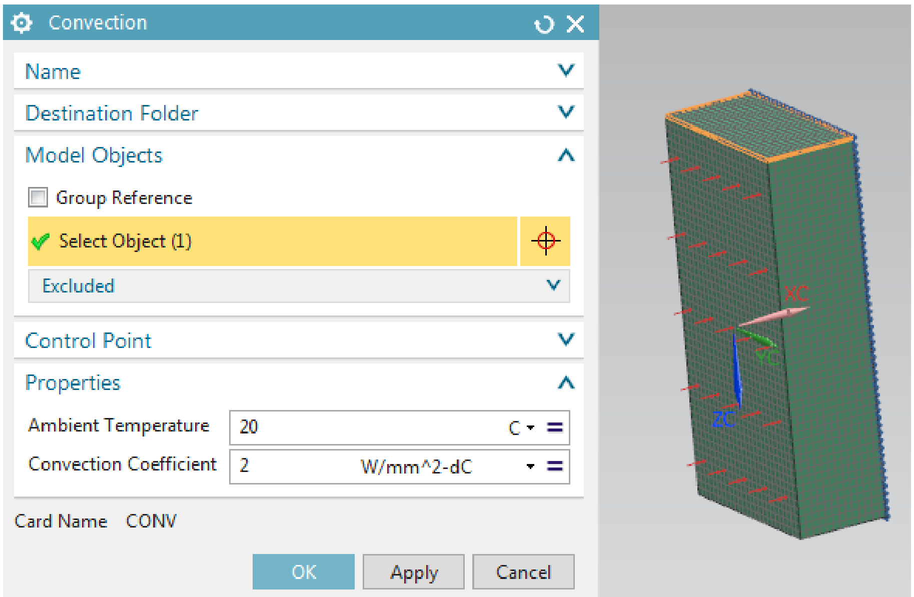


Picture 38. Thermal constraint type



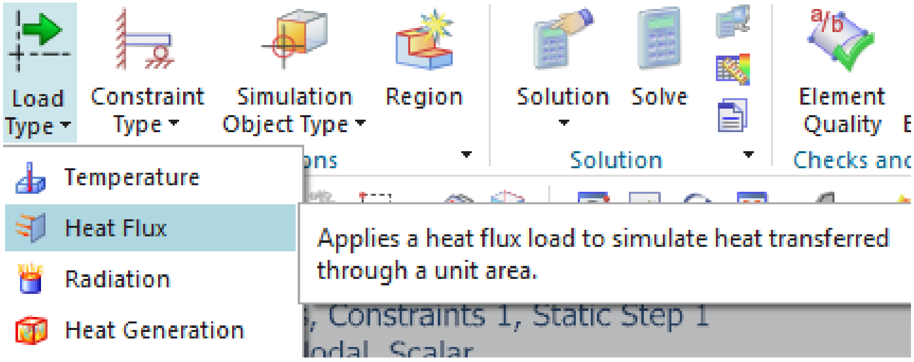
Picture 39. Thermal constraints

The second restriction is a convection area. For that must be selected convection on constrain type (picture 38). As with the fixed temperature, it is going to be chosen the face affected by the convection and consequently, the magnitude of the ambient temperature and the convection coefficient which are going to be 20ºc and 2 W/mm^{2}dC (picture 40).



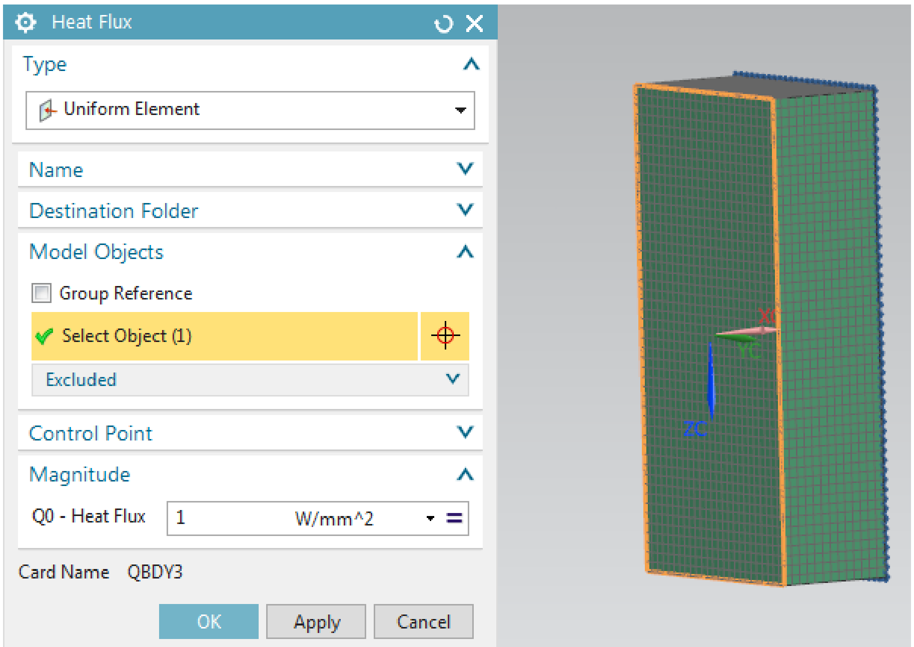
Picture 40. Convection parameters

Now it is needed to add the heat flux. For that, it is need to go on load type from the toolbar and select heat flux (picture 41)



Picture 41. Thermal load type

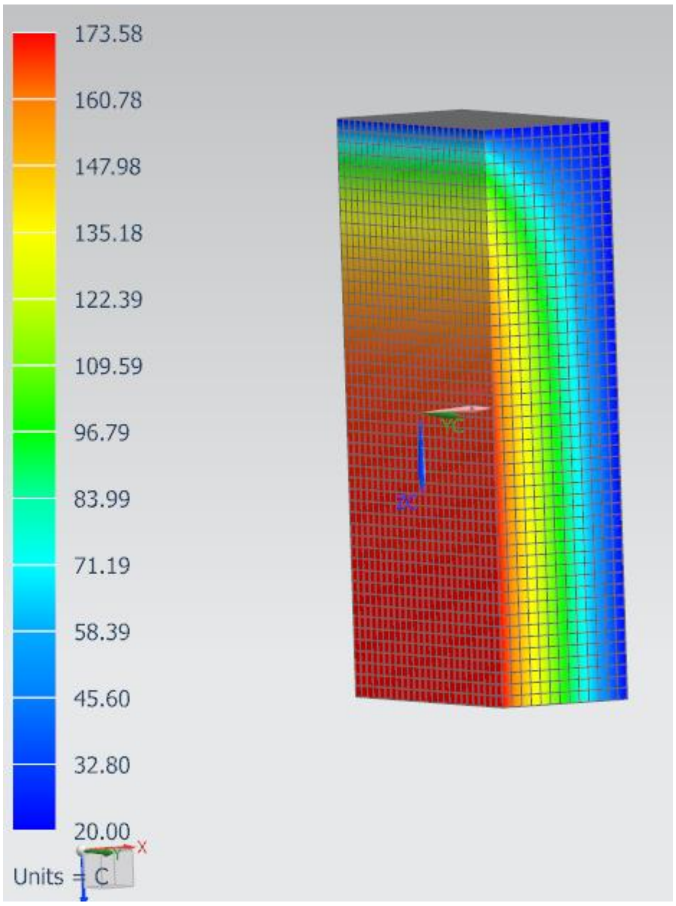
Now, as all the other restrictions, it is going to be chosen the face where the heat flux is penetrating and a 1W/mm^2 of heat flux magnitude (picture 42).



Picture 42. Heat flux

Finally, all the parameters of the problem are defined in NX. That means, it is time to solve it. The steps to order NX solving the problem are exactly the same as it has done with the structural analyzing only changing the kind of solutions. In the list of solutions, the only considerable to study is the internal variation of temperature from the wall.

<u>Internal temperature variation:</u>



Picture 43. Internal change of temperature

That solution represents the internal variation of temperature in ºC. The temperature is represented by the same range of colors from structural analyzing. The redder the temperature is, the highest it is. As it is logic, the highest temperature is found on the face the heat flux is penetrating and in the opposite edge from the convection constrain. This has a maximum valor of 173.58ºC and the more close it get to the constrains, more close to 20ºc it is.

5 SUMMARY AND CONCLUSIONS

Finally, once the thesis is over, is time to start with the final conclusions where is going to be revised the results to see if the objectives were completed and how far has the project gone.

Before starting with the analyzing of the results, it is important to remind which the goal of this project was. After TAMK received the license for NX, it was necessary to create some resource in order students could get familiarized with the basic tools easier instead of losing some valuable time during the courses. As NX is such a complex software, it was necessary to choose what it was more important to learn at first and it was decided to focus on structural and thermal analysis as they are the most common and general problems an engineer will have to solve using this kind of software. The way chosen to show how theses analysis simulation works was creating two basic problems, one for each type of analysis, and explaining how to solve them using an instruction format.

About the results, I strongly believe both examples chosen for the explanation totally hit the mark. They were structured step by step, always followed by pictures to make sure the students do not get lose in the procedure, and sometimes with some connotations explaining different ways in order to show all the possibilities NX has. And always at the end there was an explanation of how to interpret the representations we got from the solutions. Also, the theoretical part was so useful in this part in order to know why the FEM representation looked that way and to understand the reason about the reaction of the material.

After all, there is no need to say the thesis was finished fulfilling the objectives set in the beginning. Now, new students who want to start using structural and thermal simulations with NX for the first time, will have this resource to use so they will get familiarized so much fast in compare if they should learned by they own. Also, is going to be helpful for the teachers as they will need less time to explain these basic tools at the beginning of the courses.

About my personal experience, would like to mention how challenge was it, mostly on the beginning, due two main reasons. The first one is the fact that I did not have any experience with this software, only some from a similar one called ANSYS, so it was totally new as well for me and that made the procedure going slowly at the start. The second one was realizing about the complexity this software was. Sometimes, it was so difficult to choose which way should be taken and organizing it on the way to write an instructions to be understood as easy as possible. Even that, once starting to get familiar with the software and starting taking decisions about the way to explain them, things started to go ahead alone and faster. Finally, I’m strongly confident that all the knowledge acquired from this project has made me a more completely engineer and it will help me in my future professional career.

To end up, I would like to thank Tamk facilities which have made my work so much easily and of course my supervisor, Petri Pohjola, for solving all my doubts and being always available, since the first day, meet with me to resolve any kind of problem I could have.

5.1 References

Web pages:

https://en.wikipedia.org/wiki/Finite_element_method

https://www.simscale.com/blog/2016/10/what-is-finite-element-method/

https://www.plm.automation.siemens.com/en/products/simcenter/3d/index.shtml

http://www.esss.com.br/blog/es/2016/01/metodo-de-elementos-finitos-que-es/

https://www.plm.automation.siemens.com/es_es/plm/fea.shtml

Slides:

https://es.slideshare.net/urzuarq/tipos-sistemas-y-esfuerzos-de-las-estructuras

https://www.slideshare.net/gilbertomireles10/mecanismos-de-transmisin-de-calor-conduccion-conveccion-radiacion

Book/pdf

https://termoaplicadaunefm.files.wordpress.com/2012/01/teorico_introduccion_a_la_t_de_q.pdf

http://www.imac.unavarra.es/web_imac/pages/docencia/asignaturas/maquinas-iti/Apunteak/Cap1.pdf

Structural analysis with finite elements: Rugarli Paolo, Civil engineering and construction materials, 2010

Analysis of a Carbon Fibre Rim: Uyttersprot Bryan, Student Formula, 2015

Video tutorial:

https://www.youtube.com/user/SiemensNxTutorials

https://www.youtube.com/watch?v=RUC1n1_2eFg

https://www.youtube.com/watch?v=FZY71dE6Tmw